Common mistakes in designing MSE wall with finite element method

Tjie-Liong Gouw
Indonesian Chapter of IGS, Indonesia

ABSTRACT: Fast computer processors have made complicated engineering computation techniques, such as finite element method, that was once require hours and hours of computing time, has now been made possible to run all the computation in less than an hour. Since the early 2000s, the availability of commercial geotechnical finite element software combined with faster, smaller, and cheaper notebook computer has made finite element method more accessible to many engineers. However, it is very often the engineers learn just how to operate the software through the tutorial manual without really understand the underlying geotechnical engineering knowledge, they do not realize the old saying: “Garbage in garbage out”. This paper, albeit may not complete, tries to elaborate the common mistakes found in applying the finite element geotechnical software in designing geosynthetics mechanically stabilized earth wall. It does not consider the construction process of the reinforced soil wall where the fill is being built up, during which most of the geogrid strain is developed.

Keywords : MSE Wall, Reinforced Soil Wall, Finite Element Method, Plaxis, Common Mistakes

1 INTRODUCTION

Prior to the change of the millennium, limit equilibrium method is the standard calculation technique used in solving many geotechnical engineering design problems. Since the early 2000s, advancement of computer technology, development of geotechnical finite element software, together with more affordable personal computers, has made many engineers start using finite element method (FEM) in solving the geotechnical design problems. However, it is often found that engineers only learn on how to operate the software through the limited information on the tutorial manual provided by the software developer. They do not really understand the underlying geotechnical engineering knowledge and rather ignorance on the adage: “Garbage in garbage out”.

One of the application of this geotechnical FEM software is to design reinforced earth wall, also known as mechanically stabilized earth (MSE) wall. Lack of knowledge often makes engineers do not understand the significant difference between limit equilibrium method and finite element method in assessing the MSE wall design, and hence make mistakes. This paper, using Plaxis version 2016 as example, highlights the common mistakes often seen in applying FEM software in designing MSE wall, particularly with geosynthetics as reinforcing elements.

2 PLANE STRAIN VS AXISYMMETRY MODEL

Though it is relatively simple concept, many practicing engineers often fail to understand the difference of plane strain and axisymmetry models. For example, the model in Figure 1 will result in a long out of plane MSE wall construction if plane strain model is adopted. On the other hand, if axysimmetry model with rotation axis on the left hand side is adopted, it will result in a circular island shape MSE wall.

The plane strain model means the strains can only take place in the xy plane. Along the longitudinal axis (out of plane direction) the strain is assumed to be zero, $\varepsilon_z = 0$. Consequently, the length of the MSE wall must be significantly larger than its width.

The axisymmetric model means the lateral, or more precisely, the radial strains of the model are equal in all direction, $\varepsilon_x = \varepsilon_z$. As the name implies the structures in the model is symmetrical along the vertical y axis and the model is rotated at the y axis, hence the model in Figure 1 results in a circular island shape MSE wall. Note: in Plaxis the rotating axis is always at the left boundary.

Of course failure in choosing the right model of plane strain or axysimmetry will lead to incorrect output.

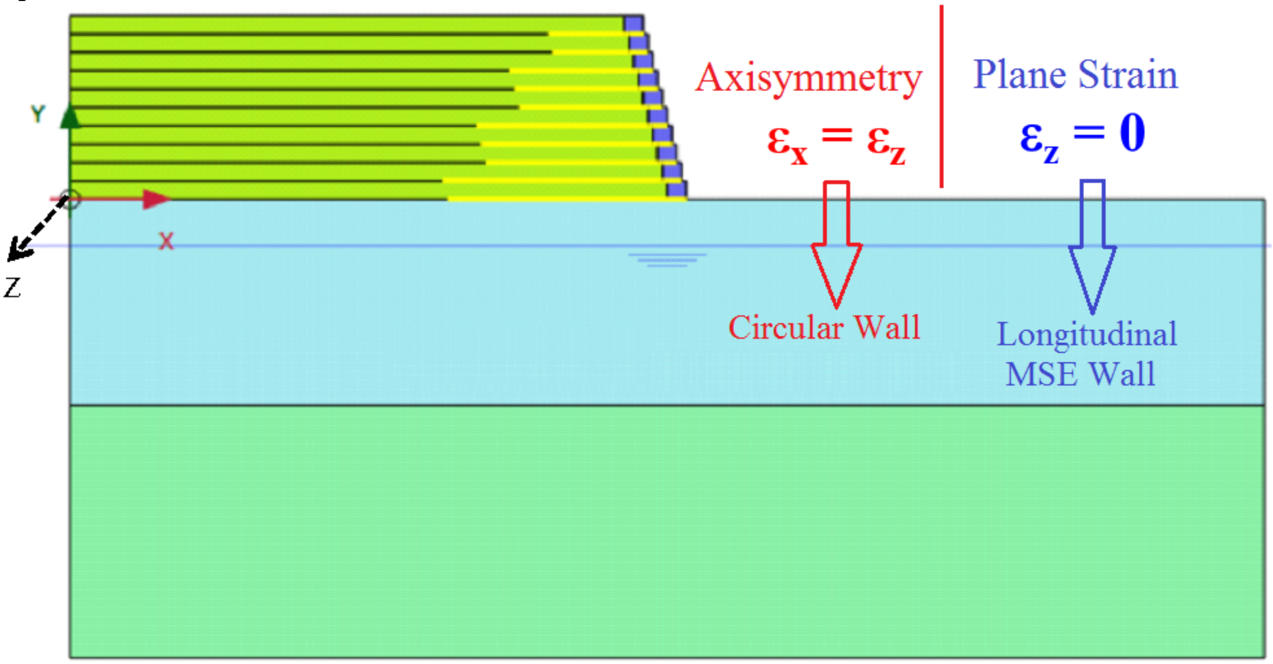


Figure 1. Plane strain vs axysimmetry model

3 MODELLING FACING ELEMENTS

The MSE walls, also known as segmental walls, are often constructed with facing elements, made of either concrete blocks or gabions. Some engineers model the facing element as plate element, in so doing the analysis will result in bending moment been induced in the facing element as shown in Figure 2.

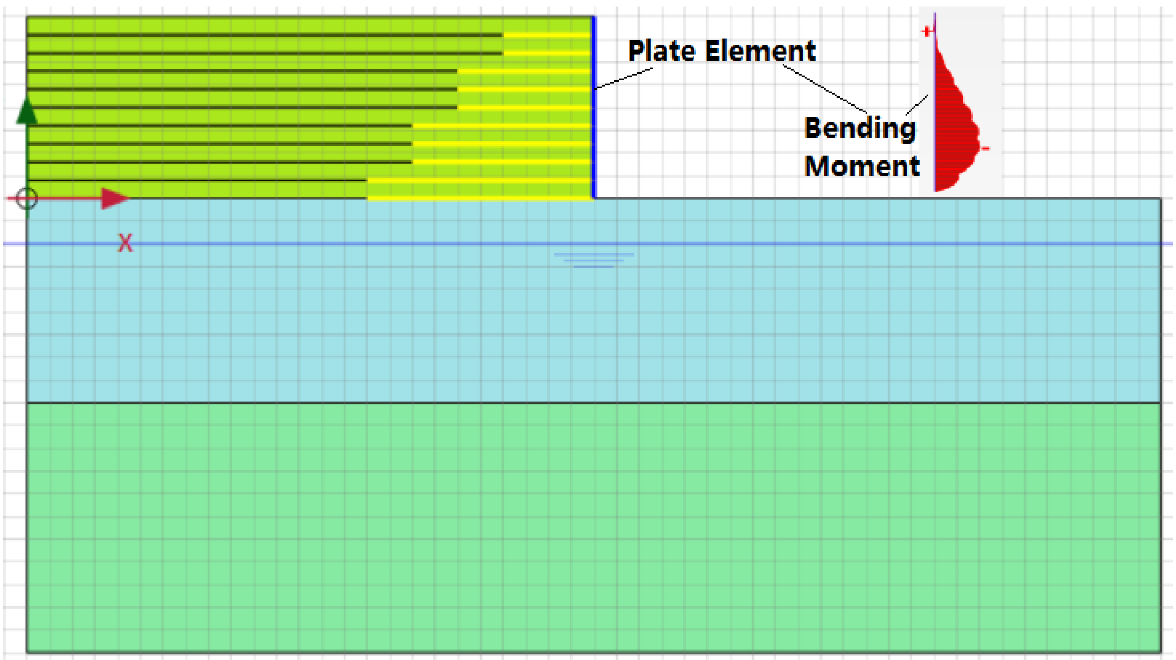


Figure 2. Model facing element as plate element will result in bending moment at facing element

In reality, there will be virtually no moment developed on the facing element. Therefore, the correct way is to model the facing element as soil cluster with dimension as the actual dimension of the facing element (Figure 3). Since the facing element is normally made of concrete blocks or gabions, which act as a block, the material model of the soil cluster can be chosen as linear elastic. If water can drain through the facing element, the drainage type is chosen as drained. If water cannot penetrate through the facing element, the drainage type is chosen as non-porous. When the facing concrete blocks/gabions can slip or move one another then 'interface elements' should be added in between the concrete blocks/gabions (see section 4.1).

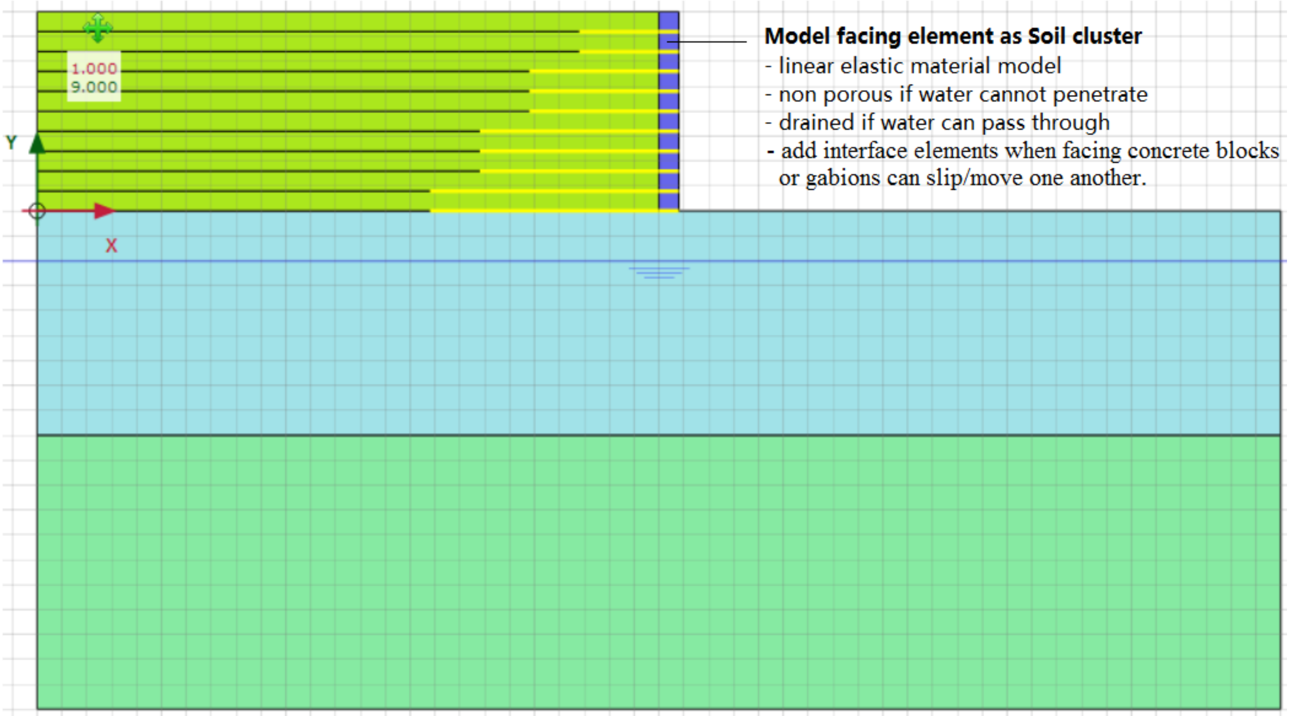


Figure 3. Appropriate way is to model facing element as soil cluster with its corresponding material properties

4 MODELLING GEOSYNTHETICS REINFORCING ELEMENTS

Geosynthetics is basically a geo-construction material that can only withstand tensile force, therefore it should be modelled as tensile element. In Plaxis this tensile element is termed as "geogrids" element. Although it is termed as 'geogrids' element, it does not mean it can only be used to model geogrids! Basically this 'geogrids' element can model any thin construction material that withstand tensile force only. Hence, it can be used to model almost all types of geosynthetics, e.g.: geotextiles, geogrids, and geomembranes. Engineers know water can pass through geotextiles and geogrids, but cannot penetrate through geomembranes. However, many does not know how to model those three different type of geosynthetics properly in a finite element analysis, as a result sometime the geogrids element drawn can actually mean geomembrane or vise-versa. To properly model these three types of geosynthetics one has to incorporate 'interface' element.

4.1 *Interface elements*

Interface element in finite element model is used to model the contact area between two types of different material, e.g., model the contact area between geosynthetics and the soil, between concrete and soil, etc. This interface element, particularly in Plaxis, has two functions. The first function is to reduce the friction between the soil and the construction material in contact with the soil by introducing an interface reduction coefficient (a value between 0 to 1). The second function is to indicate whether the interface is impermeable or permeable. Figure 4 shows how to properly model geotextile, geogrids and geomembrane by combining 'geogrids' and 'interface' elements.

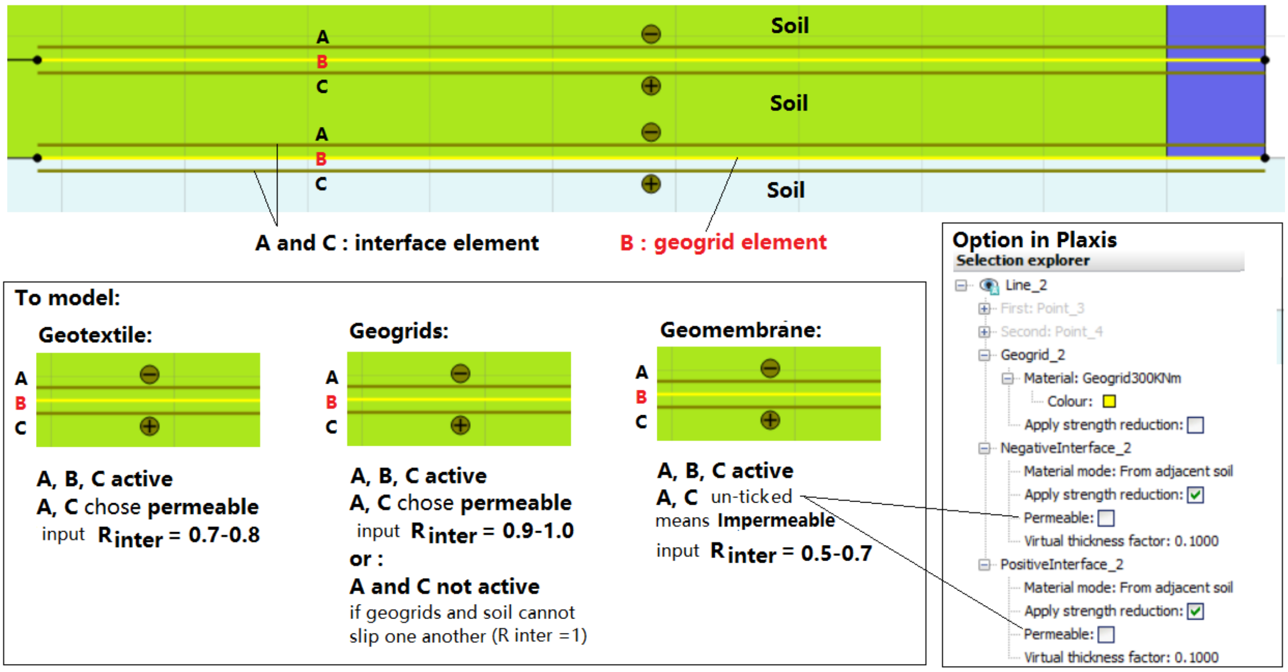


Figure 4. Interface element to model geotextile, geogrids, and geomembrane

The first function of applying 'interface' element to 'geogrids' element, is to assign a maximum possible friction between the soil and the geosynthetics. This is done by introducing an interface interaction coefficient, known as R_{inter}, in Plaxis, with a certain value. The program then calculates the friction of the soil and the geosynthetics as presented in Equation 1 and Equation 2.

$$c_{geosynthetics} = R_{inter} \text{ x } c_{soil} \tag{1}$$

$$(\tan \phi)_{geosynthetics} = R_{inter} \text{ x } (\tan \phi)_{soil} \tag{2}$$

The value of R_{inter} depends on the roughness of the surface area of the geosynthetics, for geotextiles R_{inter} normally within 0.7 to 0.8, for geomembrane can be as low as 0.5 to 0.7. For geogrids, unlike geotextile and geomembrane where the pull out resistance only depends on friction between their surface and the soil, owing to its structural shape which composes of longitudinal and transversal elements with opening in between the two elements, in addition to surface friction, passive resistance also generated through the contact of the soil particle with its transversal element as illustrated in Figure 5. Therefore, the R_{inter} value of geogrids can be as high as 0.9 to 1.0. A value of 1 means the soil particles and the geogrids will move together as if it is one unity and allowing no slippage between the soil and the geogrids. For this condition, the geogrids element can also be modelled without interface elements. It is suggested to adopt the R_{inter} value from pull out test results carried out on the type of geosynthetics under consideration. Note that MSE wall construction normally use geogrids as reinforcing elements.

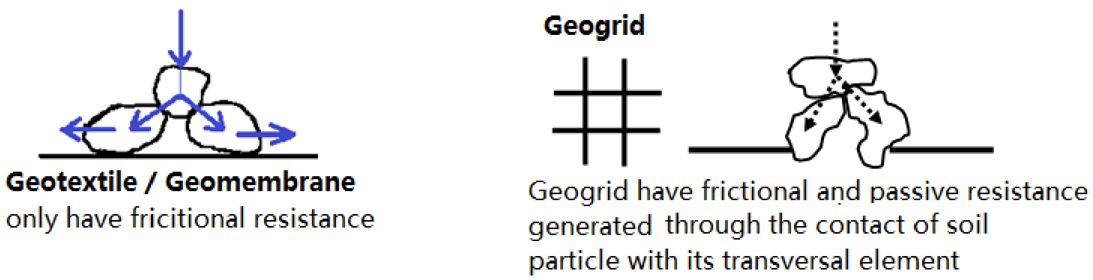


Figure 5. Frictional resistance between soil and geotextile, geomembrane, or geogrids

The second function of applying 'interface' element to 'geogrids' element, is to indicate (to tell the software) whether the modelled geosynthetics is permeable or impermeable. As water can pass through geogrids and geotextiles, the 'interface' element must be indicated (chosen) to be permeable. On the other hand, as water cannot pass through geomembrane then it must be indicated as impermeable, by not selecting it at the 'permeable' option in Plaxis as shown in Figure 4. Note that in the earlier version of Plaxis, e.g. v8 and v9, the reduction of frictional resistance is introduced by activating the 'interface' element in soil mode; the impermeability of the 'geogrids' element is introduced by activating the 'interface' element in water mode, non-active 'interface' element in water mode means water can flow through the 'geogrids' element.

4.2 *Axial stiffness, EA, of geosynthetics*

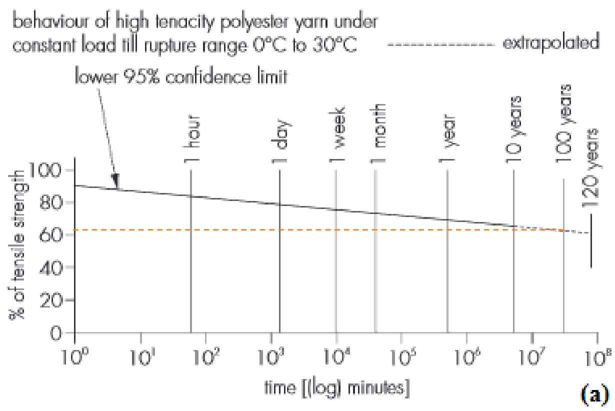
Another important factor that commonly misunderstood is the input of axial stiffness of 'geogrids' elements. Many engineers often simply adopt the breaking strength (in kN per m run) provided in the technical brochure as the axial stiffness of the selected geosynthetics material. This is definitely an erroneous approach! The determination of this axial stiffness is given below. Detail explanation can be seen on the author's previously published paper (Gouw, 2015).

Before the stiffness of the geosynthetics material can be determined, the design strength of the geosynthetics, T_{all}, to be used as reinforcing element has to be calculated by using Equation 3 (Koerner, 2005, Sarsby, 2007).

$$T_{all} = \frac{T_{ult}}{RF_{CR}\, x RF_{ID}\, x RF_{CBD}\, x RF_{JOINT}} \tag{3}$$

where: T_{ult} = short term ultimate (breaking strength)
RF_{CR} = reduction factor due to creep
RF_{ID} = reduction factor for installation damage
RF_{CBD} = RF_{CD} x RF_{BD}
RF_{CD} = reduction factor for chemical damage
RF_{BD} = reduction factor for biological damage
RFjoint = reduction factor for joints/seams

The value of the short term ultimate strength can be obtained from the breaking strength provided by the manufacturer of the geosynthetics. The reduction factor due to creep is determined through time creep degradation curve or isochronous curve of the relevant material, Figure 6 shows an example of isochronous curve for Polyethyline (PET) material (Chamberlain & Cooper, 2008). If the design life of the MSE wall is 100 years, and when the strain is not a limiting factor, then from Figure 6a, the remaining strength after 100 years is around 62% of the breaking (ultimate) tensile strength. This means RF_{CR} = 1/0.62 = 1.62. If strain is limiting factor, then use Figure 6b, if the limiting strain is 5%, after 100 years the remaining strength is 33%, it means RF_{CR} = 1/0.33 = 3.



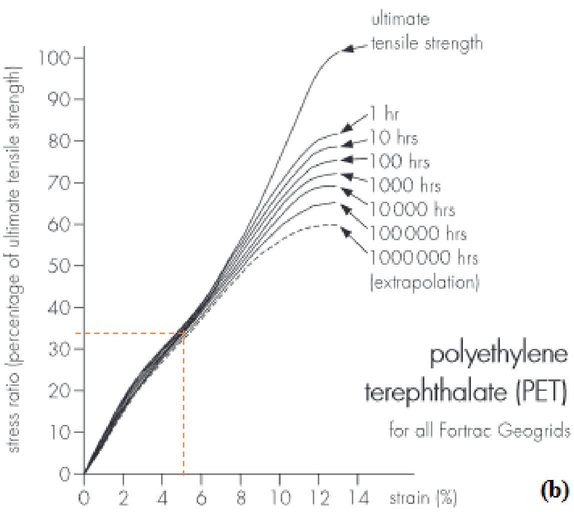


Figure 6. Time Creep Degradation & Isochronous Curve (Chamberlain & Cooper, 2008)

Other reduction factors can be obtained from Table 1 (Koerner, 2005). The values of RF_{CR} in the table shall be used only when there is no creep reduction or creep isochronous curve available. The low end of the RF_{CR} range refers to applications which have relatively short service lifetimes and/or where creep deformations are not critical to the overall performance of the geosynthetics reinforced structures.

Table 1. Geosynthetics Strength Reduction Factors for Reinforced Soils (extracted from Koerner, 2005)

Application	RF_{CR}	RF_{ID}	RF_{CBD}
MSE walls	2.0 – 4.0	1.1 - 2.0	1.0 – 1.5
Embankments	2.0 – 3.5	1.1 - 2.0	1.0 – 1.5
Bearing and foundations	2.0 – 4.0	1.1 - 2.0	1.0 – 1.5
Slope Stabilization	2.0 – 3.0	1.1 – 1.5	1.0 – 1.5

When there is joint in the geosynthetics reinforced soil structures, joint reduction factor, RF_{JOINT}, values can be taken within 1.8 to 2.0. However, it is best to use one piece of geosynthetics without any joint.

Once the allowable tensile strength of the geosynthetics is determined, its stiffness can be calculated as follows:

$$E = \frac{\text{Stress}}{\text{Strain}} = \frac{\sigma}{\varepsilon} = \frac{T_{all}}{A} \cdot \frac{1}{\varepsilon} \rightarrow \quad EA = \frac{T_{all}}{\varepsilon} \tag{4}$$

where: E = Young's modulus
σ = Stress
ε = Strain
T_{all} = Allowable tensile strength
A = cross sectional area of geosynthetics in used
EA = tensile stiffness of geosynthetics

Since the allowable tensile strength is normally stated in load per unit width (kN/m) and the strain is dimensionless, the tensile stiffness, EA, obtained is also in unit of load per unit width (kN/m). Hence, the axial stiffness of geosynthetics is obtained by dividing its allowable tensile strength with its corresponding allowable strain. For reinforced soil structures where deformation should be limited, the author suggests the limiting strain presented in Table 2 (Gouw, 2015).

Table 2. Limiting Strain of Geosynthetics for Reinforced Soil Structures (Gouw, 2015)

Application	Limiting Strain (%)
MSE walls	3 - 5
Embankments	6
Bearing and foundations	2
Slope Stabilization	4 - 5

Example of the geosynthetics axial stiffness and short term capacity calculation is given below:

- Given breaking strength of a geocomposite is, T_{ult} = 300 kN/m.
- For slope stabilization, design life 100 years
- Limiting strain = 5%
- RF_{CR} = 1/(33%) = 3 (from Figure 6b)
- RF_{ID} = 1.10
- RF_{CBD} = 1.04
- No joint
- T_{all} = 300 / (3x1.10x1.04) $\approx$ 88 kN/m
- EA = 88 / 5% $\approx$ 1760 kN/m

4.3 *Elastic or elastoplastic material?*

Apart from determining its tensile stiffness, the geosynthetics material behavioural type has to be determined, whether it will act as elastic or elastoplastic material. When the material type is chosen to be elastic, it means there will be no limit to its strength. For high MSE walls or high slope stabilization applications, rather than modelling the geosynthetics as elastic material, it is better to model it as elastoplastic material so that the tension force acting at the geosynthetics layers can be limited up to the allowed long term capacity derived as per Equation 3. Continuing from above example, then the limiting tensile force shall be:

- $T_{all\text{-}long\ term}$ = 300 / (3x1.10x1.04) = 88 kN/m (case with no geosynthetics joint is allowed in MSE wall)

5 SOIL PARAMETERS

Like in any other geotechnical analysis, a reasonably accurate input of soil parameters is very important, otherwise, the calculation shall be as good as the old adage which says: "Garbage in garbage out." Some notes on the effect of soil parameters on the design of MSE wall are elaborated here.

5.1 *Backfill material parameters*

Non cohesive granular material is a preferred material to be used as backfill to build MSE walls. Here in Indonesia, lateritic cohesive clay material is also often used as backfill material. To construct a reliable MSE wall, this lateritic cohesive material must be compacted under drained condition (compaction under undrained condition is not recommended as it reduces the stability of the MSE wall and also increases its long term deformation). However, one must make sure not to over-estimate its cohesion parameter. The basic formula for calculating lateral earth pressures acting on a retaining wall requires the input of the soil cohesion, as presented in the equations below:

$$P_a = k_a\, \sigma_v' - 2c' \sqrt{k_a} \qquad (5)$$

$$P_p = k_p\, \sigma_v' + 2c' \sqrt{k_p} \qquad (6)$$

where: P_a = active earth pressure,
P_p = passive earth pressure,
k_a = active earth pressure coefficient,
k_p = passive earth pressure coefficient,
σ_v' = effective overburden pressure, and
c' = drained cohesion.

From the above formula, it can be seen that the larger the c' value, the lesser the active earth pressure, on the other hand, the passive earth pressure increases. It is clear that, over estimating c' will lead to unsafe condition. Hence, it is important to properly test the shear strength parameters of this compacted backfill material.

5.2 *Foundation soil undrained parameters*

When the MSE wall is built on top of saturated clay foundation soil, undrained analysis must be performed. The soil mechanics lesson normally told us that analysing undrained behaviour of clay has to be done with total strength parameters, S_u or c_u, ϕ'=0, undrained or total stiffness parameters, E_u, and undrained Poisson's ratio, υ=0.5. However, in many FEM codes, the undrained analysis is often calculated through effective stress approach. The reasoning behind is: there is a mathematical relationship between undrained and drained shear strength parameters as shown in Figure 7 (Gouw, 2014), where the undrained shear strength parameter can be related to the drained (effective) parameters as presented in Equation 7.

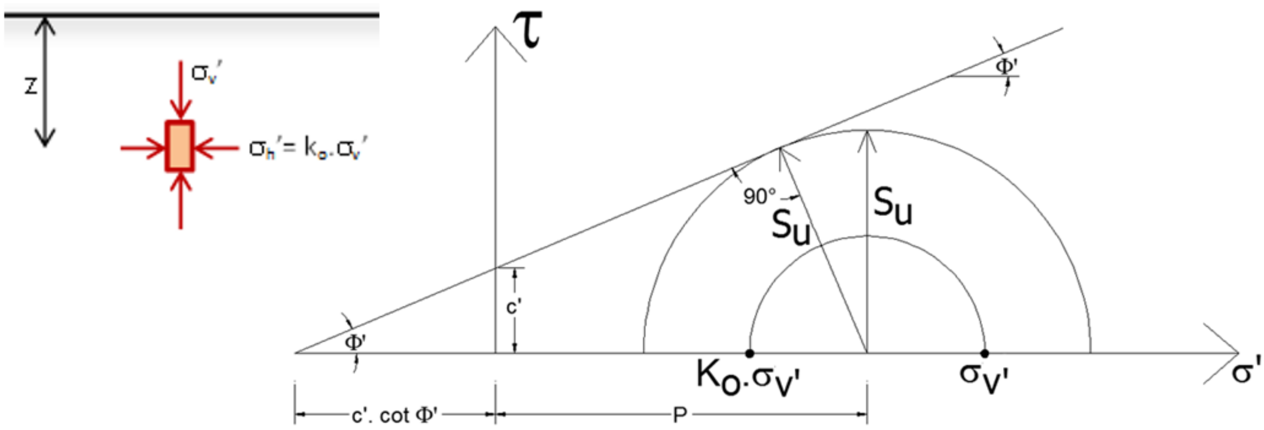


Figure 7. Effective Stress Formulation of Undrained Strength (Gouw, 2014)

$$C_u = Su = Sin\, \emptyset'.\left(c'.\cot \emptyset' + \frac{K_0.\sigma_{v'} + \sigma_{v'}}{2}\right) \tag{7}$$

where: $C_u = S_u$ = undrained shear strength (undrained cohesion),
c' = effective (drained) cohesion,
ϕ' = effective (drained) angle of internal friction,
K_0 = at rest earth pressure coefficient,
σ_v' = effective overburden pressure, and

In Plaxis, there are three combination of inputs in modeling the undrained shear strength, as presented in Table 3.

Table 3. Modeling Undrained Analysis

Undrained type	Parameters
Undrained A	Analyzed in term of effective stress. Pore pressure generated. Material type: undrained Effective strength parameters c´, φ´, ψ´ Effective stiffness parameters E_{50}´, ν´
Undrained B	Analyzed in term of effective stress Pore Pressure generated but NOT accurate Material type: undrained Total strength parameters c= c_u φ=0, ψ=0 Effective stiffness parameters E_{50}´, ν´
Undrained C	Analyzed in term of effective stress Pore Pressure NOT generated Material type: drained / non porous Total strength parameters c= c_u φ=0, ψ=0 Total stiffness parameters E_u, ν_u = 0.495

Plaxis automatically adds stiffness of water when undrained material type is chosen, therefore, if total stiffness parameters are adopted as taught in the conventional soil mechanics, then the undrained stiffness becomes very much higher than it should be. In turn, it will lead to inaccurate predicted deformation.
Since soil behavior is always governed by effective stresses, Undrained A is a preferable method in modeling undrained behavior of clay. It can predict the excess pore water pressure in a relatively accurate manner, and increases of shear strength during consolidation can be calculated. However, here, the undrained shear strength is a calculated (not an input) parameter, therefore, care must be taken if Mohr Coulomb soil model is adopted as undrained A may over predicts the undrained shear strength (see Figure 8).

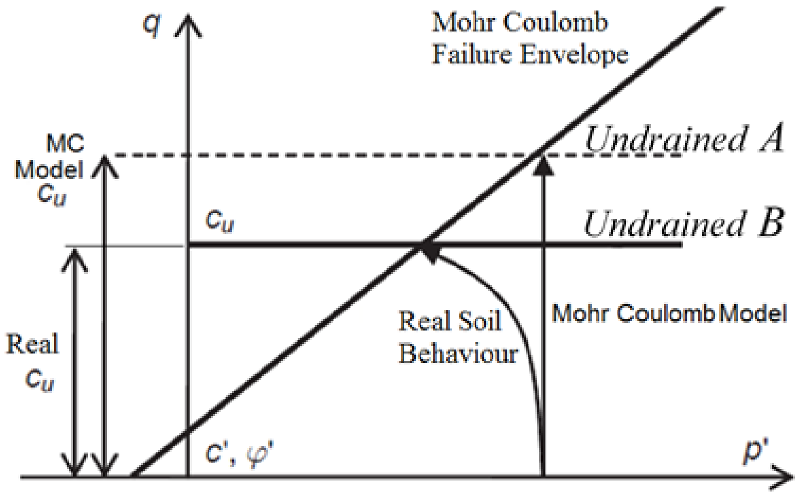


Figure 8. Effective Stress Formulation of Undrained Strength (Gouw, 2008)

Undrained B and undrained C options lead to a more accurate short term overall stability of the MSE wall, since the undrained shear strength is limited to the input parameter. However, it cannot be used to calculate consolidation and subsequent increases in undrained shear strength.

6 INITIAL STRESS OF FOUNDATION SOILS

One of the major differences with limit equilibrium method is finite element analysis take into consideration the initial stress (in situ stress) of the foundation soils.

Initially, when creating the finite element model, although the soil parameters has been assigned and the finite element mesh has been created, the soil body self-weight, i.e. the initial stresses, has not been counted for. A special procedure is necessary to generate or to calculate the initial stresses within the soil body. As the name implied, initially only the original soil body exists, therefore, all the structural elements and geometry changes, e.g.: backfilling, excavation, all structural elements of the MSE wall must not be activated.

At this stage, engineers, very often, again without understanding the proper theoretical knowledge, directly go through the so called k_o procedure, to generate the initial water pressure and the initial effective stresses of the original ground. The k_o procedure, calculates the stresses within the soil body by the following simple equation:

$$\sigma'_{ho} = k_o \sigma'_{vo} \tag{8}$$

where σ_{ho}' is the horizontal earth pressure at rest, k_o is the coefficient of earth pressure at rest, σ_{vo}' is the effective vertical overburden pressure. This procedure is correct only and only when all the geometry of the ground surface, the ground layers, and the ground water table are horizontal (Figure 9).

Where the ground surface, the subsoil layer, or the ground water level is not horizontal, as shown in Figure 10, the k_o procedure will lead to the existence of unbalance forces or non-equilibrium of initial forces within the soil body, which are obviously not correct. In such cases, to maintain equilibrium, there should be shear stresses developed within the soil body. Therefore, the k_o procedure should not be used, instead a *gravity loading procedure*, where the shear stresses are calculated should be chosen.

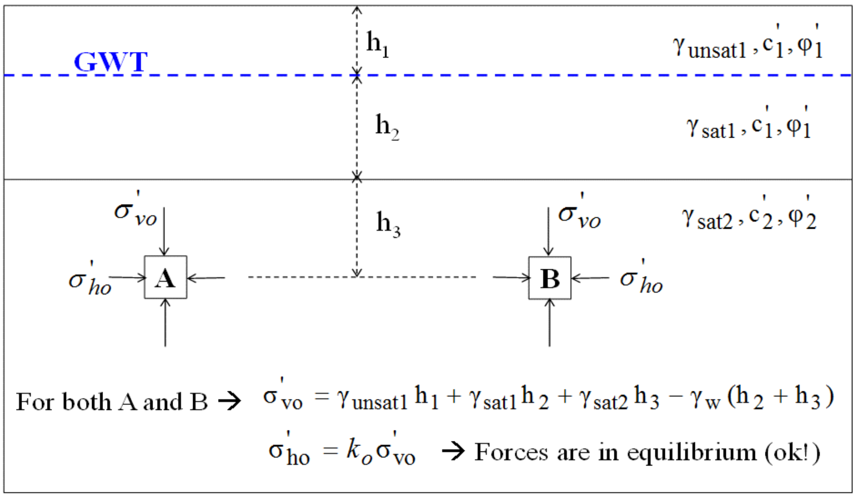


Figure 9. K_o Procedure for Horizontal Geometry

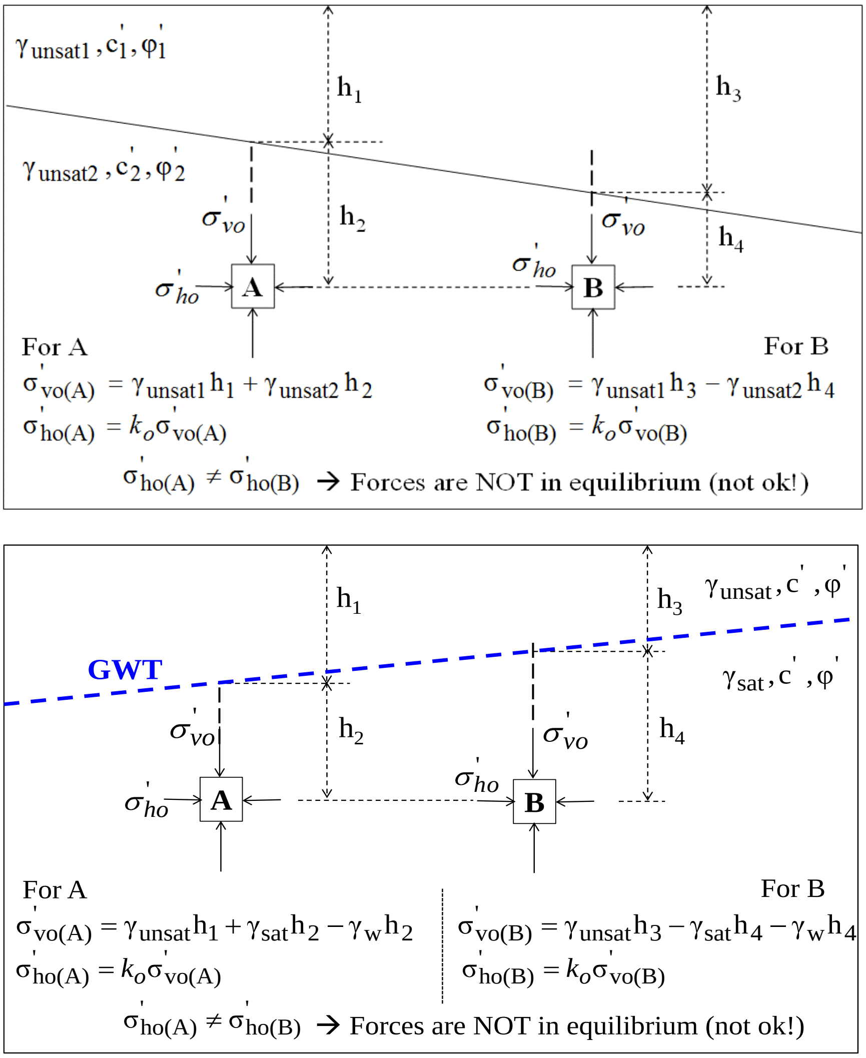


Figure 10. Cases where K_o Procedure is Inaccurate

The option of gravity loading and k_o procedure in the initial phase is available in Plaxis 2D version 2011 and above. For Plaxis 2D version 9 and below, the gravity loading stage needs to be done by skipping the k_o procedure. This is done by setting ΣMweight=0 in the k_o procedure i.e. in the initial stage. This way no initial stresses within the soil body is developed. The initial stresses of the soil body are then calculated in the calculation module of the program by selecting the first phase as plastic 'Calculation type', and if any of the soil layer is modeled as undrained, the 'Ignore undrained behavior' option in the 'Parameter' tab has to be selected (this is due to the fact that initially, when no external load and no geometry changes is made, the soil is in a drained condition). In the 'Loading input' section, the 'Total multiplier' option is selected, and in the 'Multiplier' tab, key in ΣMweight=1. Then the next actual construction stages are modeled.

7 FACTOR OF SAFETY ANALYSIS

In limit equilibrium analysis, the stability of MSE walls must be analysed in three parts. The first part is the internal stability of the MSE structure itself, whether the geosynthetics have adequate pull out and breaking resistance against the acting forces. The second part is the external stability, i.e. whether the MSE structures as a block, have the required translational, rotational, and bearing capacity factor of safeties. The last part is the global stability, i.e. the stability against sliding.

Engineers often ask, how to obtain all such factor of safeties with finite element software. The answer is: Finite element method can only give one safety factor, the weakest one among all those generated by limit equilibrium calculation. The reason is, generally in finding the factor of safety, finite element software does not carry out the step by step safety analysis such as in the limit equilibrium software. It carries out the safety analysis by keep on reducing the shear strength of the soil, until a chain of soil elements reach plastic point (failure point) is formed and failure is triggered. The safety factor is then obtained by dividing the original shear strength parameter by the last shear strength parameter that trigger failure. The formulation is presented in Equation 9.

$$FS = \frac{c_o}{c_f} = \frac{\tan\phi_o}{\tan\phi_f} \tag{9}$$

Where c_o is the initial cohesion, c_f is the cohesion that trigger failure point, ϕ_o is the initial friction angle, ϕ_f is the friction angle that trigger failure point. This generally means the other safety factors shall be larger than the weakest safety.

8 CONCLUDING REMARKS

It is important to highlight here that the above write up does not consider the construction process of the MSE wall where the fill is being built up and during which time most of the geogrid strain is developed. The paper highlights some of the common mistakes and misconception often encountered in analysing the stability of MSE walls with finite element method. As whether the assessment written here will reasonably predict the long term deformation of the MSE wall, it needs to be investigated further.

The advance of computer technology and the availability of the FEM geotechnical software provide engineers with sophisticated tools for analyzing geotechnical problems on hand. However, without a proper training, this sophisticated tools can mislead the engineers using it. They might think their analysis is correct since they have used state of the art software, without realizing they have made mistakes, where serious mistake can lead to a catastrophic end of the project on hand. The author himself, along the years of learning and applying the geotechnical FEM software, has made many mistakes. Finally, the success of analyzing MSE wall with FEM greatly depends on good understanding of soil mechanics, the soil behavior and its relevant parameters, the structural properties, and also on the background of the software on hand.

REFERENCES

Chamberlain, B. and Cooper, G. (2008), BBA Agreement Certificate no 05/4266 for Fortrac Geogrids.

Gouw, T.L. (2014), Common Mistakes on the Application of Plaxis 2D in Analyzing Excavation Problems, International Journal of Applied Engineering Research, Research India Publications, ISSN 0973-4562 Volume 9, Number 21 (2014), pp. 8291-8331.

Gouw, T.L. (2015), Importance of Elongation Factor in Determining Geosynthetics Stiffness for Finite Element Calculation, Proceedings of International Conference on Landslides and Slope Stability, Bali, Sep 27-30, 2015, pp.B8-1 to B8-6.

Koerner, R.M. (2005), Reduction Factors Used in Geosynthetics Design, GSI White Paper 4, GII Publications, Folsom, Pennsylvania, 13 pp.

Sarsby, R.W. (2007), Geosynthetics in Civil Engineering, Woodhead Publishing Limited, Cambridge, England.